Pro/ENGINEER® 2001

Pro/WELDING™ Topic Collection

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About Pro/WELDING

Pro/WELDING is an optional Pro/ENGINEER module that allows you to model welds in assemblies. In addition, you can generate report tables about weld parameters and show welding symbols in assembly drawings.

Using Pro/WELDING, you can:

- Create and modify welds in Assembly mode.
- Blank or display welds in an assembly.
- Define parameters of the welding process.
- Create assembly drawings with welding symbols.
- Obtain information about welds (such as, mass, volume, size, and so forth).
- Generate Pro/REPORT tables with rod and weld information.

About a Weld Feature

Welds are created in a top level assembly as an assembly feature.

A weld feature does not change geometry of welded components. Before you proceed to welding, make sure components to be welded have appropriate profiles.

A weld is modeled as a quilt. Adding a weld does not merge referenced components. When you retrieve a component that was welded in an assembly, its geometry remains the same.

Weld features are parametric and associative. If referenced geometries change, a weld feature (such as, its location, length, volume, and so on) also updates.

The following types of weld beads can be created in Pro/WELDING:

- Fillet
- Butt or Groove
- Plug
- Slot
- Spot

In addition to simple welds, you can also create compound welds, including "both sides" welds and reinforced welds.

A weld feature has the following characteristics:

- Feature ID
- Weld sequence ID
- Welding rod
- Welding parameters
- Weld type
- Geometric references

Each weld is identified by its name in the following format:

:*Type_of_ weld*, Rod: *Rod_name*

where:

#— is the sequence ID of the weld.
Type_of_ weld—is the type of weld (such as, "Fillet Weld").
Rod_name—is the name of a rod (such as, "Steel_001").

For example:

```
1:Fillet Weld, Rod: Rod1
```

Supported Weld Types

Pro/WELDING allows you to create simple and compound welds.

The following types of simple welds are supported in both ANSI and ISO standards:

Fillet	
Butt or Groove	
Square Butt or Groove	
V-Butt or V-Groove	
Bevel Butt or Bevel Groove	
U-Butt or U-Groove	
J-Butt or J-Groove	
Flare V-Butt or Flare V-Groove	
Flare Bevel Butt or Flare Bevel Groove	
Plug	
Slot	
Spot	

To Create a Weld

1. Enter Assembly mode and retrieve or create an assembly.
2. Click **Applications > Welding**. The **WELDING** menu appears.
3. Define the welding environment by assigning the rod and defining welding parameters.
4. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
5. Select the type of weld and define the weld. Click *See Also* for more details.
6. Create drawings of welded assemblies and annotate weld joints.
7. Generate either a BOM or Pro/REPORT tables with weld parameters.

Finite Element Analysis of Welded Assemblies

You can perform a Finite Element Analysis (FEA) of an assembly with weld joints using the same techniques as with other assemblies. A variety of FEA elements can be used to model weld elements, as for example, bar elements, shell elements, and contacts.

To analyze a FEM model and review the results of analysis, you need licenses for Pro/MESH and Pro/FEM-POST.

About Welding Rods

A welding rod provides the welding material necessary to create a weld bead. A welding rod has a circular cross-section with a user-defined diameter.

A welding rod can be used for a single or multiple welds.

Within your welding assembly, each rod is defined by its name and parameters associated with this rod. Rod parameters are stored with the model. If you want parameters of a particular rod to be used for defining welds in other assemblies, you can store the rod parameters to a file on disk.

To Create a Welding Rod

You can create a welding rod or read in a welding rod parameter file using the **WELDING RODS** dialog box.

1. Click **Welding > Rod**. The **WELDING RODS** dialog box opens.
2. Click **File > New**. Enter the name of the welding rod then click  . The name of the welding rod appears in the rod list.
3. Set the welding rod parameters.
4. Determine optional and user defined parameters.
5. Click **Apply.** Pro/Welding issues a message indicating the welding rod with the specified name has been successfully created.

Welding Rod Parameters

The following table lists parameters that can appear in the rod parameter file. The default parameter values are shown in *italics*. Parameters shown in bold have negative default values which you must substitute with user-specified values.

Rod Parameter Table

Parameter Name	Value	Definition
DIAMETER	*-1.000*	Rod diameter
LENGTH	*-1.000*	Length of the rod
DENSITY	*-1.000*	Density of the rod material
SPECIFICATION_NUMBER	*1EXXXXX* string	Specification number of the rod
MATERIAL	*CAST_IRON* string	Material of the rod
LENGTH_UNITS	*INCH* FOOT MILLIMETER CENTIMETER METER	Units of length for the rod
MASS_UNITS	OUNCE *POUND* TON GRAM KILOGRAM TONNE	Units of mass for the rod
USER_DEFINED	string	A user-defined parameter can be added to the list of parameters.

When you define parameters in the Pro/TABLE window, you can include additional parameters or specify user-defined parameters with the F4 key. Press F4 and a parameter dialog box appears with a list of parameters from which to choose. You can also choose parameter values with the F4 key.

To Save Welding Rod Parameters

You can save welding rod parameter files using the **WELDING RODS** dialog box.

1. Click **Welding > Rod**. The **WELDING RODS** dialog box opens.
2. Click **File > Save**. The contents of a rod parameter file is saved to the disk. Pro/Welding appends the extension "rod" to the file name. The rod parameter file is now independent from and not associative with the rod.

To Rename a Welding Rod

1. Choose **Rod** from the **WELDING** menu.
2. Choose **Rename** from the WELD ROD menu to change the name of the welding rod.

To Read in a Welding Rod File

1. Click **Welding > Rod**. The **WELDING RODS** dialog box opens.
2. Click **File > Open**. The **Open** dialog box opens.
3. Select the name of the welding rod file from the Model Tree menu. The system reads in the specified file and creates a welding rod with the name of the file. If one of the existing rods has the same name, the system asks you if you want to overwrite the existing rod. If you answer [N], enter a different name for the new rod.

To Delete a Welding Rod

1. Click **Welding > Rod**. The **WELDING RODS** dialog box opens.
2. Select the rod name from the Rod List.
3. Click **File > Delete**. If this rod was used for other welds in the assembly, Pro/Welding requests a confirmation before the rod is deleted. You need to assign a new rod to those welds that used the deleted rod.

To Assign a Welding Rod to a Weld

1. Click **WELDING > Rod**.
2. Click **File > New** to create a new rod

...or...

Click **File > Open** and select one of the existing rods by choosing its name from the **OPEN** dialog box.

3. Click **Utilities > Assign**. The **Names** dialog box opens.
4. Select the names of welds to assign, then click **OK**. The **GET SELECT** menu appears.
5. Click **Done Sel**.

To Define Welding Parameters

For each weld, you must specify relevant welding parameters. You can use previously defined welding parameters.

Click **WELDING > Parameters**. The following commands are listed in the **WELD PARAMS** menu:

- **Retrieve**—Retrieve a welding parameter file such as, *filename*.wpr from disk. If a welding parameter file already exists in the assembly, the system asks you if you want to overwrite the existing parameter file.
- **Save**—Store a parameter file to disk. Enter the name for the file. The system appends the extension ".wpr" to this file and stores it in the current directory.
- **Mod Params**—Modify parameters by editing a list of welding parameters in the Pro/TABLE environment.
- **Show**—Show the current welding parameters in an info window.

Welding parameters can be defined in one of the following ways:

- Edit the default (or current) list of welding parameters using the **Mod Params** command in the WELD PARAMS menu.
- Retrieve an existing welding parameter file from disk with the **Retrieve** command.

Welding Parameters

Parameter Name	Value	Definition
X_SECTION_AREA	value *-1.000*	Cross-sectional area of the weld
ROOT_PENETRATION	value *0*	Depth of root penetration
MAX_ALLOWED_LENGTH	value *1000*	Maximum allowed length of the weld
MIN_ALLOWED_LENGTH	value *0*	Minimum allowed length of the weld
MAX_ROOT_OPENING	value *100*	Maximum root opening
MIN_ROOT_OPENING	value *0*	Minimum root opening
SPECIFICATION_NUMBER	string *1EXXXXX*	Specification number of the weld
TREATMENT	NONE *LOW_HYDROGEN* PRE_HEATING POST_HEATING	Treatment of the weld
FINISH	CHIP *GRIND* HAMMER MACHINE ROLL UNSPECIFIED	Finish process of the weld
MACH_TYPE	*MANUAL* ROBOTIC	Machining type of the weld
FEEDRATE	value *-1.000*	Feedrate of the welding machine (unit = assembly unit /hour)
LENGTH	value	(Read-only) Computed length of trajectory of the welding rod
WELD_LENGTH	value	(Read -only) Computed length of the (physical) weld bead.
GROOVE_ANGLE	value *0.000*	Angle of a groove weld between welded components
LEG1	value *0.000*	Value given to the first leg of a fillet weld

LEG2	value *0.000*	Value given to the second leg of a fillet weld
PLUG_SIZE	value *0.000*	Size of the plug weldment
PREP_DEPTH	value *0.000*	The preparation depth of a weld
ROOT_OPEN	value *0.000*	The size of the root opening between two welded components
SPOT_PITCH	value *0.000*	The pitch distance between spot welds
USER_DEFINED	string	A user-defined parameter can be added to the list of parameters

About Welds

Welds are parametrically defined features; they are associative with the referenced geometry and can be manipulated as other standard Pro/ENGINEER features. Weld faces are represented in the assembly as quilts.

To Create a Fillet Weld

You can create continuous or intermittent fillet welds. You can also locate the ends of the fillet weld with respect to the geometry of welded components.

1. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
2. After you set up your welding feature, combination, and environment, click  in the **WELD DEFINITION** dialog box.
3. Type the fillet weld dimension for L1 (Leg 1). If you want to define unequal lengths for each leg, type the length dimensions for L1 and L2 (Leg 2) and clear the **Equal Length** check box.
4. Click the **Optional and User Defined Parameters** arrow to add or delete welding parameters.
5. Click **OK**. The **Menu Manager** dialog box opens.
6. Specify reference entities using options in the **REF OPTIONS** menu. Choose **Edge-Edge** or **Surf-Surf** and specify reference geometries.
7. Choose an entity with the options on the **Get Select** menu. If you choose **Sel By Menu**, a list of parts is displayed from which you can choose an entity.
8. Choose the **Done Sel** option from the **Get Select** menu.
9. When you finish defining references for each component, choose **Done Refs** from the **Feature Refs** menu.
10. Choose **Edge-Edge** to specify continuous preparation edges.
11. Specify placement constraints of the weld using options in the **Placement** menu.
12. Click **OK**.

Fillet Weld Reference Information

Pro/ENGINEER calculates the fillet weld trajectory for two adjacent surfaces by projecting the edge of each surface on the other.

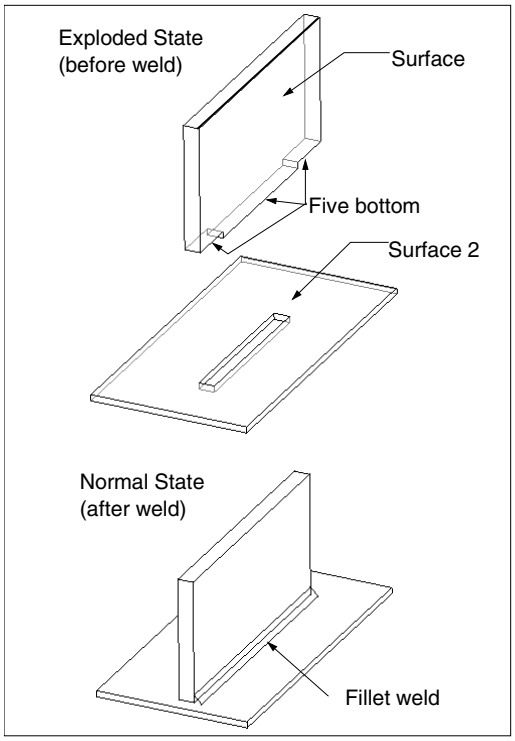
Prior to Release 20, the trajectory could only be one continuous line. In effect, this meant that the two surfaces had to make contact only where one edge of one surface continuously contacted the other surface (although not necessarily along the full length of the edge).

You can weld surfaces in which the weld trajectory is broken into two or more continuous segments. These cases can be broken down into two classes, as shown in the following examples:

- Segmented trajectory; one physical weld.

  Example: two plates in a "tap and slot" construction, secured by a single fillet weld. There is continuous metal-to-metal contact between the two surfaces along the desired weldpath. See the following figure. In this case, the algorithm automatically creates a single weld trajectory corresponding to the physical path.

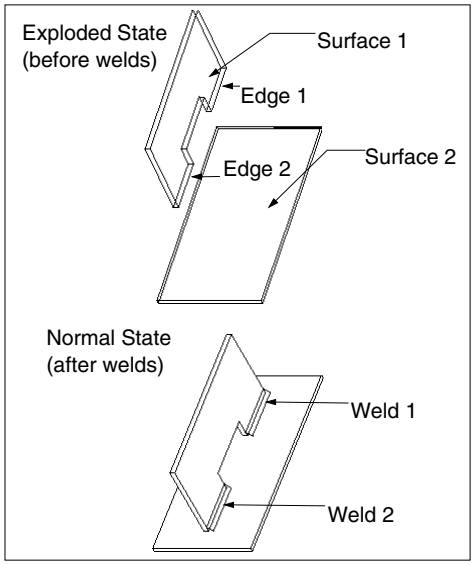
Fillet Weld of Two Plates Joined with Tap-and-Slot Construction



- Segmented trajectory; multiple physical welds.

  Example: two plates which make contact along several discrete edges. Each such contact would correspond to a separate physical weld. See the following figure.

Fillet Weld of Two Plates, one of which has a Cutout on the Weld Edge



In this case, the algorithm realizes that the surfaces are contacting along several discrete edges, that there is not continuous metal-to-metal contact and that the surfaces could be joined by several discrete welds. Since it can only create one weld at a time, it asks you to resolve the ambiguity. It does so by displaying a Resolve Ambiguity menu.

The menu items are named Tr1, Tr2, and so on, where each item corresponds to a trajectory segment on the model. As you move the cursor over a particular menu item, the corresponding trajectory segment is highlighted on the model.

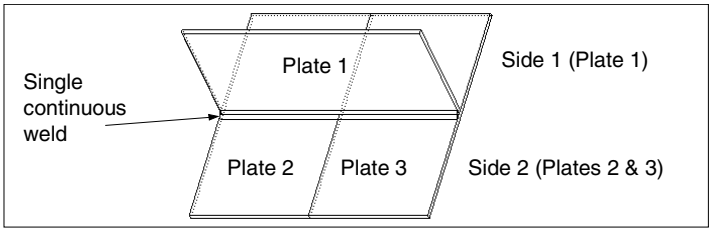
Select one entry, then choose **Done Sel**. After you define the Placement, the Fillet Weld dialog box will then update to show the new element "Ambiguity". At this stage, all the elements will be defined and you need only choose **OK** to create the weld.

Creating Surface-Surface Fillet Welds Across Multiple Components

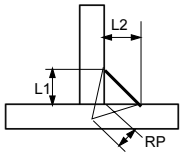
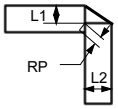
Prior to Release 20, it was not possible to create a single surface-surface fillet weld that extended across more than two components. That restriction no longer applies.

In the following figure is an example of a simple surface-to-surface fillet weld that joins one plate to two others. As before, you can pick more than one continuous surface for each side of the weld. However, those surfaces can now be on adjacent assembled components.

One Plate Fillet Welded to Two Other Plates.



About Fillet Welds

Type of Weld	Example	Key Dimensions
Fillet (T joint)		L1 - first leg distance L2 - second leg distance RP - root penetration
Fillet (corner joint)		L1 - first leg distance L2 - second leg distance RP - root penetration

To Specify Reference Geometries

When creating a weld, you must specify the location of the weld by referencing geometry elements of the components to be welded. For this, you use options in the Ref Options menu.

Note: Depending upon the weld type, the Ref Options menu lists only those options that are valid for the particular type of weld.

The REF OPTIONS menu provides the following options:

- **Edge-Edge**—A weld is created by referencing edges of two components. For each component, you need to specify edges using options in the Chain menu. Once you have selected a reference edge or a chain of edges for the first component, choose Done Sel. If needed to adjust the selected edges, use the Trim/Extend option. To conclude the selection for the first component, choose Done from the Chain menu. Do the same for the second component.
- **Surf-Surf**—A weld is created by referencing surfaces of two components. For each component, you must specify continuous surfaces using options in the Feature Refs menu. Choose Add and start selecting continuous surfaces from the first component. When finished, choose Done Refs. Do the same for the second component.
- **Curve-Curve**—A weld is created by referencing datum curves of the components to be welded. For each component, specify a datum curve or a chain of curves using options in the Chain menu. You can use either One By One to select individual curves or Curve Chain to select a chain of curves by picking a curve from the chain. Once the reference curve or chain of curves is defined for the first component, choose Done Sel. If needed to adjust the selected curves, use the Trim/Extend option. To conclude the selection for the first component, choose Done from the Chain menu. Do the same for the second component.
- **Edge-Surf**—A weld is created by referencing a chain of edges of one component and continuous surfaces of the other component. A chain of edges is selected as with the option Edge-Edge, and surfaces are selected as with the option Surf-Surf.

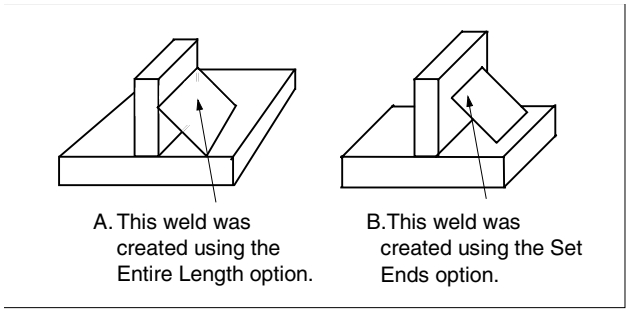
To Specify Placement Constraints

For fillet welds, placement constraints are specified using the following options in the PLACEMENT menu:

- **Entire Length**—Create a weld along the entire length of matching geometries.
- **Set Ends**—Set the ends of the weld using the following options in the SET ENDS menu:

  - **Mod End**—Specify the location of the ends of the weld. The system highlights one end of the weld's trajectory. Choose Accept or Next to indicate the end to be set. Drag the selected end to its new location and dimension it using options in the END DIM TYPE menu. If necessary, set the other end in the same manner.
  - **Start Point**—Set one end of the weld to be the start point. This option allows you to set the direction for creating welds.
- **Continuous**—Create a continuous weld.
- **Intermittent**—Create an intermittent weld.

Using the Entire Length and Set Ends Options



END DIM TYPE Menu Options

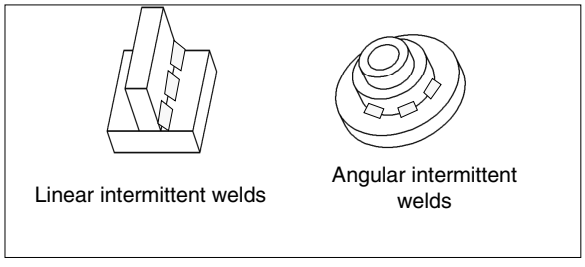
When setting the ends of the weld, the END DIM TYPE menu appears with the following options:

- **Ext Length**—Establish the extension distance by entering the length of section. Notice that when the system prompts you to enter the length value, the current length is indicated in brackets.
- **Offset Plane**—Establish the extension distance by measuring from a selected plane. A direction arrow displays the positive extension direction. You can align the end of the weld with a selected plane by specifying the zero offset value.
- **Offset Csys**—Establish the extension distance by measuring from a selected coordinate system.

To Specify Intermittent Welds

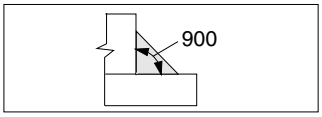
Intermittent welds can be linear or angular, as shown in the following figure. Specify the desired type by choosing **Linear** or **Angular** from the Placement menu.

Linear and Angular Intermittent Welds



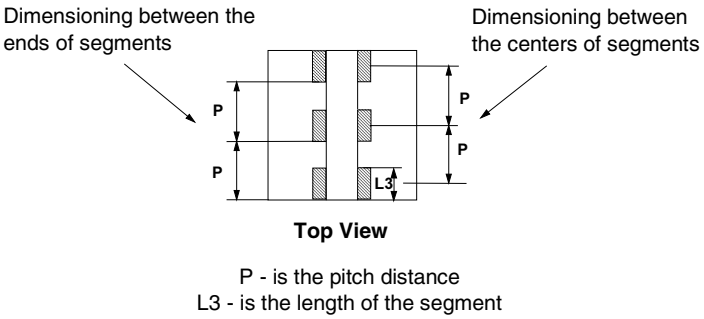
Note: Angular intermittent welds are supported only for cylinder surfaces which are perpendicular to the corresponding welded surface.

Example of Angular Intermittent Weld



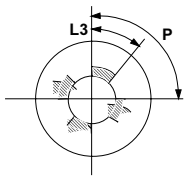
Segments of an intermittent weld can be dimensioned between the centers or between the ends. Specify the desired dimensioning scheme by choosing **At Center** or **At End** from the Spacing menu.

Dimensioning a Linear Intermittent Weld



P - is the pitch distance
L3 - is the length of the segment

Dimensioning an Angular Intermittent Weld



Top View

P - is the pitch distance in degrees
L3 - is the length of the segment in degrees

To Place an Intermittent Weld

1. From the PLACEMENT menu, choose:
   - **Entire Length** or **Set Ends**
   - **Intermittent**
   - **Linear** or **Angular**
   - **Done**
2. If you chose **Set Ends**, set the end of the first weld segment.
3. Enter the length of the segment. For angular intermittent welds, this value is entered in degrees.
4. From the SPACING menu, make the following selections:
   - Specify the dimensioning scheme by choosing **At Center** or **At Ends**.
   - Specify the number of welds by selecting one of the following options:
   - **Pitch Dist**—Enter the pitch distance for the intermittent weld. For angular welds, enter the pitch distance in degrees.
   - **Num of Welds**—Enter the number of welds.
   - Choose **Done**.
5. Enter a value (that is, the pitch distance or the number of welds) as prompted by the system.
6. Continue the procedure as for continuous fillet welds.

To Route a Butt or Groove Weld

1. Click **Create > Welding**. The **WELD DEFINITION** dialog box opens.
2. Set up your feature, combination, and welding environment.
3. Click the desired type of groove in the **WELD DEFINITION** dialog box. Click *See Also* for more details.
4. Type the measurements for the root opening, preparation depth, penetration, and angle dimensions in

the text boxes.
5. Click the **Optional and User Defined Parameters** arrow to add or delete welding parameters.
6. Click **OK**.
7. Choose an entity with the options on the **Get Select** menu, then click **Done**. If you choose **Sel By Menu**, a list of parts is displayed from which you can choose an entity.
8. The **REF OPTIONS** and **CHAIN** menus appear. You must specify both geometry elements of the components to be welded and chain options, then click **Done**. Click *See Also* for more details.
9. Click **OK**.

About Butt or Groove Welds

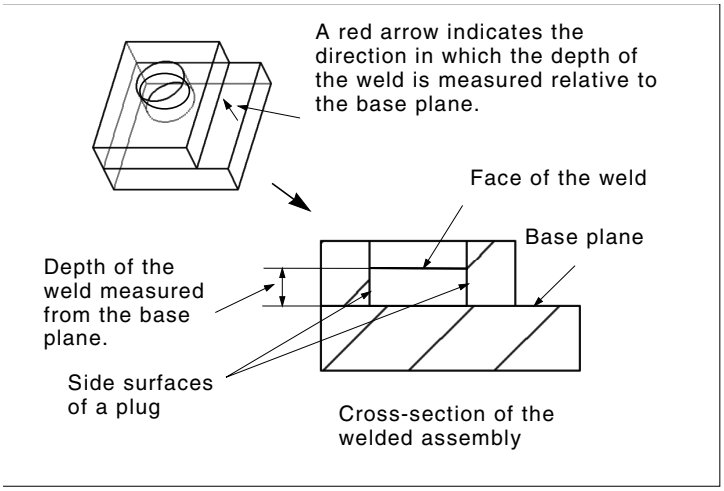
Groove Weld Types

ANSI	ISO
Groove	Butt
Square Groove Double Square	Square Butt
Single V-Groove Double V-Groove	Single V-Butt Double V-Butt
	Single V-Butt with Broad Root Face Double V-Butt with Broad Root Face
Single Bevel Groove Double Bevel Groove	Single Bevel Butt Double Bevel Butt
	Single Bevel Butt with Broad Root Face Double Bevel Butt with Broad Root Face
U-Groove Double U-Groove	Single U-Butt Double U-Butt
J-Groove Double J-Groove	Single J-Butt Double J-Butt
Flare V-Groove Double Flare V-Groove	
Flare Bevel Groove Double Flare Bevel Groove	

To Route a Plug and Slot Weld

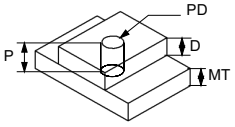
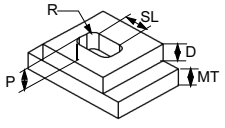
When defining a slot/plug weld, you need to set the base plane from which you must measure the depth of the weld so you can locate the face of the weld.

Creating a Plug/Slot Weld



1. Set up the welding environment.
2. Choose **Plug** or **Slot** from the WELD ROUTE menu.
   A dialog box appears, listing elements of the weld feature that need to be defined. They are:
   - **Plug/Slot Refs**—Specify geometric references for the weld.
   - **Base Plane**—Specify the base plane.
   - **Depth**—Specify the penetration depth.
3. The system brings up the FEATURE REFS menu, allowing you to select side surfaces of the plug/slot. When finished selecting surfaces, choose **Done Refs** from the FEATURE REFS menu.
4. Specify the base plane for the plug/slot weld using options in the PLANE menu:
   - **Plane**—Select a planar surface or a datum plane to be the base plane.
   - **Make Datum**—Create a datum plane to be the base plane.
   - **Quit Plane**—Quit specifying the base plane.
5. Enter the depth of the weld, measured from the base plane.
6. Select the direction for measuring the depth from the base plane. A red arrow appears, pointing from the base plane in the direction of measuring. Choose **Flip** or **Okay**.
7. If necessary, define measurement parameters.
8. To conclude feature creation, choose **OK** from the dialog box.

About Plug/Slot Welds

Type of Weld	Example	Key Dimensions
Plug		P – penetration/height D – depth of a hole PD – plug diameter MT – material thickness
Slot		P – penetration/height D – depth of a slot MT – material thickness SL – slot length R - slot radius

To Route a Spot Weld

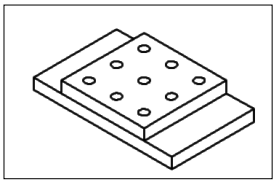
Spot welds are located by referencing datum points. You can reference existing datum points or create datum points in the process of routing the weld.

There are two methods of creating a spot weld:

- Define multiple locations for spot welds and create multiple welds in one operation.
- Create a single spot weld and pattern it with the Pattern option.

Spot welds are displayed as circular surfaces. The diameter of the circle is calculated by the system based on the value of the X_SECTION_AREA parameter entered by you when defining weld parameters. To modify the size of the spot weld, you need to change the X_SECTION_AREA parameter. You can set up a relation controlling the area of the weld.

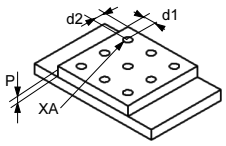
A Spot Weld



1. Set up the welding environment.
2. Choose **Spot** from the WELD ROUTE menu.
3. A dialog box appears, listing elements of the weld feature that need to be defined. They are:
   - **Spot Refs**—Specify geometric references for the weld.
   - **Penetration**—Specify the penetration depth.
   - **Measurements**—Create measurements used to control welding parameters.
4. Locate the weld by referencing datum points. Create or select datum points for locating the weld using options in the SEL POINT menu. Choose **Create** and create datum points, or **Select** and pick existing datum points.
5. When finished specifying reference points, choose **Done** from the FEATURE REFS menu.
6. Enter the penetration distance.
7. If necessary, define measurement parameters.
8. To conclude feature creation, choose **OK** from the dialog box.

Once a single spot weld is created, you can pattern it with the Pattern option in the Weld Utils menu.

About Spot Welds

Type of Weld	Example	Key Dimensions
Spot		P - penetration/height XA - cross-section area d1, d2 – dimensions locating the center of the spot weld R - radius of the spot weld is calculated as follows: $R = \sqrt{XA/\Pi}$

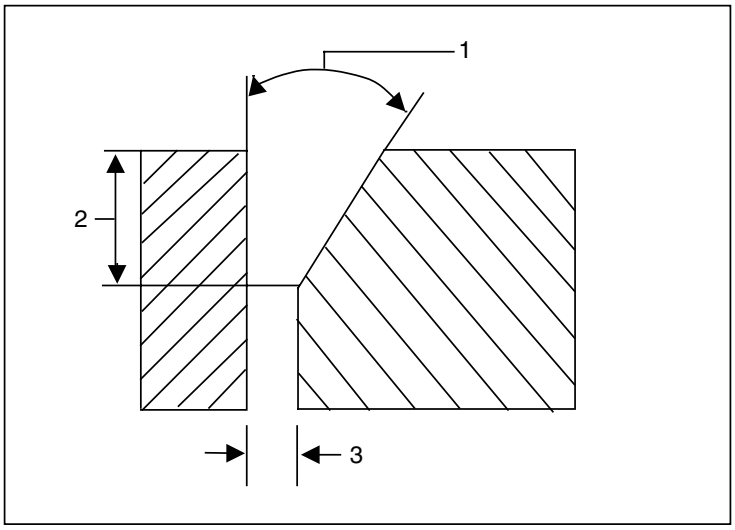
To Prepare Edges for Welding

1. Click **Create** from the **WELDING** Menu. The **WELD DEFINITION** dialog box opens.
2. Click the **Environment** arrow to display the types of edge preparation cuts. Click *See Also* for more details.
3. Click the type of edge preparation cut you want to use.
4. Type the measurements for the root opening, preparation depth, penetration, and angle dimensions in the text boxes.
5. Select **Create Family Table Instance** or leave the check box clear if you want to create the cut on the generic part and assembly. Click *See Also* for more details.
6. Click **Part** or **Assembly** to establish feature dependency.
7. Click **OK**. The **SURF OPTIONS** menu appears. Click *See Also* for more details.
8. Select an option and then select the surface that you want to prepare for welding.
9. Click **Done Sel**.

About Edge Prep Dimensions and Display Options

The **WELD DEFINITION** dialog box allows you to declare default values for depth, groove angle, and root opening for weld preparation.

Examples of Depth, Groove Angle, and Root Opening



1. Groove Angle
2. Depth
3. Root Opening

The Family Table provides the functionality to create the cut in either the generic part or instances of the part and its assemblies. In addition, you have the option of not creating instances of the part or assembly. Click the **Environment** arrow in the **WELD DEFINITION** dialog box to access the **Create Family Table Instance** check box and **Edge Prep Driven by** options. Click the **Part** or **Assembly** button to establish feature dependency.

Three configuration settings provide the flexibility to customize instance naming conventions and specify on which models the edge preparation features are located.

Configuration Setting	Default	Description
weld_edge_prep_instance	no	Controls whether family table instances are created. If set to yes, instances of the parts, assemblies, and subassemblies of parts that receive edge preparation are created.
weld_edge_prep_name_suffix	ep	The part name and the extension make the instance name.
weld_edge_prep_visibility	generic	This option applies only if weld_edge_prep_instance is set to *yes*. Edge preparation features are resumed in the generic part and suppressed for instances of the part. The opposite applies when *instance* is chosen.

Note: If weld_edge_prep_instance is set to yes, weld_edge_prep_visibility is set to instance, and the instance assembly is not active in any window, a new window appears. You can add additional edge preparation features in that window.
The default options are set so that you can observe the edge preparation feature being applied. These features can exist at the part or assembly level, depending on desired behavior. Specify if you want these features to be family table instances or not.

To Choose Operations for Edge Preparation Features

1. Click **WELDING > Prep Utilities**. The **PREP UTILITIES** menu appears with the following options:
   - **Delete—**Delete the edge preparation feature.
   - **Redefine—**Redefine the edge preparation feature.
   - **Mod Prep Dim—**Modify the edge preparation feature dimension.
   - **Suppress—**Suppress the edge preparation feature.
   - **Resume—**Resume the edge preparation feature.
2. Click the type of operation to perform on the edge preparation feature.
3. Select the edge preparation feature.
4. Click **Done Sel**.

To Create a Root Opening

Create a root opening to offset contact surfaces.
1. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
2. Click **Edge Prep Only** and the desired edge preparation combination.
3. Click the **Environment** arrow. The **Environment** dialog box opens.
4. Click  for a one-side root opening or  for both sides.
5. Type the root opening dimension.
6. Select **Create Family Table Instance** or leave the check box clear if you want to create the cut on the generic part and assembly. Click *See Also* for more details.
7. Click the **Part** or **Assembly** button to establish feature dependency.
8. Click **OK**. The **Surf Options** menu appears. Click *See Also* for more details.
9. Select an option, the surface that you want to cut, and then **Done Sel**.
10. Click **OK**.

To Create an Angle Cut

Create an angle cut to offset contact surfaces.

1. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
2. Click desired feature and edge preparation combination.
3. Click the **Environment** arrow. The **Environment** dialog box opens.
4. Select **Create Family Table Instance** or leave the check box clear if you want to create the cut on the generic part and assembly. Click *See Also* for more details.
5. Click the **Part** or **Assembly** option button to establish feature dependency.
6. Choose the type of cut to create (bevel, V-butt or V-groove).
7. Type the measurements for the root opening, preparation depth, penetration, and angle dimensions in the text boxes.
8. Click **OK**. The **Surf Options** menu appears. Click *See Also* for more details.
9. Select an option and then select the surface that you want to cut.
10. Click **Done Sel**. The **Chain** menu appears. Click *See Also* for more details.
11. Choose the type of chain to use, select the edge to prepare, and then click **Done**.
12. Click **OK**.

To Create a Root Opening and Angle Cut Combination

Create a root opening and angle cut combination to offset contact surfaces.

1. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
2. Click the desired feature and edge preparation combination.
3. Click the **Environment** arrow. The **Environment** dialog box opens.
4. Select **Create Family Table Instance** or leave the check box clear if you want to create the cut on the generic part and assembly. Click *See Also* for more details.
5. Click **Part** or **Assembly** to establish feature dependency.
6. Choose the type of root opening and angle cut to create:
   - **V-butt or V-groove—**Prepares the weld surface with a V-butt or V-groove cut and root opening.
   - **Bevel—**Prepares the weld surface with a bevel cut and root opening.
7. Type the measurements for the root opening, preparation depth, penetration, and angle dimensions in the text boxes.
8. Click **OK**. The **Surf Options** menu appears. Click *See Also* for more details.
9. Select an option and then select the surface that you want to cut.
10. Click **Done Sel**. The **Chain** menu appears. Click *See Also* for more details.
11. Choose the type of chain to use and then click **Done**.
12. Click **OK**.

About Edge Preparation

Edge preparation consists of removing material along edges of metal surfaces. You must cut the edges of the metal in a particular way so that full welding penetration is achieved. The weld replaces the removed material and makes a complete bridge between the joining parts. You must prepare edges for welding when parts and assemblies require certain strength. Edge preparation is only possible on certain weld types. Root opening preparation is available for bevel, V, J, U and square butts or grooves. Angle cut preparation is available only for bevel, V, and square butts or grooves.

If a gap exists between two surfaces, the dimension of the gap are considered in the equation when you specify root and angle cut measurements.

Butt or Groove Weld Types

The following types of butt or groove welds are listed in the **WELD DEFINITION** dialog box.

- Square
- V
- Bevel
- U
- J
- Flared V
- Flared Bevel

Edge Preparation Cut Types

The following types of edge preparation cuts are listed in the **WELD DEFINITION** dialog box:

- One-side root opening
- Both-sides root opening
- Bevel-groove angle cut
- V-groove angle cut
- Bevel-groove angle cut with root opening
- V-groove angle cut with root opening

Chain Menu

During edge preparation and welding you need to select a chain of edges using options in the **CHAIN** menu. You must select the chain type and pick the defining entities. The **CHAIN** menu options are as follows:

- **One By One—**Define a chain by selecting individual edges and curves, including composite curves, one at a time. You can select the edges or curves in any order.
- **Tangnt Chain—**Define a chain by selecting an edge, including all of the edges tangent to this edge.
- **Bndry Chain—**Define a chain by selecting a quilt and using its one-sided edges. If the quilt has more than one loop, select a specific loop to define the chain.
- **Surf Chain—**Define a chain by selecting a surface and using its edges. If the surface has more than one loop, select a specific loop to define the chain.
- **Select—**Include all the highlighted edges.
- **Unselect—**Cancel the previous set of selections.

Surface Options

During edge preparation, you need to select surface options to perform operations. The **SURF OPTIONS** menu options are as follows:

- **Indiv Surfs—**Select surfaces that act as bounding surfaces.
- **Surf & Bnd—**Select surfaces by defining a seed surface and bounding surfaces. The **Seed Surface** option in the **Surf & Bnd** menu is active by default so that you can select the seed surface. After the seed surface is specified, you must define bounding surfaces using the **Boundary** option in the **SURF&BND** menu. Bounding surfaces are not included in the set. Choose a desired method for defining the surface boundaries from the **Bnd Method** menu:

  Indiv Surfs—Select surfaces that act as bounding surfaces.
  Loop Surfs—Select bounding surfaces by defining loop surfaces. Pick a surface to define a

loop. If the selected surface contains more than one loop, you are prompted to select an edge that indicates the desired loop. All surfaces that lie along this loop are the bounding surfaces.

- **Solid Surfs—**Select the surface of a part or assembly component to copy.

About Weld Definition

You can create welds and prepare edges using the **WELD DEFINTION** dialog box. A typical Pro/WELDING session might include the following steps. After reviewing the following steps, click *See Also* for more details.

1. Import the reference part into the welding environment.
2. Determine if you want to weld only, weld and prepare edges, or prepare edges only.
3. Define the type of weld or edge preparation to perform on the part or assembly.
4. Determine the family table configuration. The family table provides the functionality to create the cut in either the generic or instances of the part and its assemblies.
5. Determine if you want your weld or feature to contain solid or lightweight geometry.
6. Type edge preparation cut or weld dimensions.
7. Define parameters of the welding process.

About Manipulating Welds

Choosing **WELD UTILS** in the **WELDING UTILITIES** menu allows you to perform the following options:

- **Pattern**—Pattern a selected weld.
- **Delete**—Delete selected welds.
- **Del Pattern**—Delete patterned welds and remove pattern definitions.
- **Redefine**—Redefine a selected weld.
- **Mod Dim**—Modify dimensions of a weld. Pick a feature to display its dimensions, select a dimension to be modified, and enter its new value.
- **Suppress**—Suppress selected welds. Select a weld to be suppressed and choose **Done** from the **SELECT FEAT** menu.
- **Resume**—Resume selected suppressed welds Choose the names of the welds to be resumed by selecting from the namelist menu.
- **Reorder**—Re-arrange the sequence of welds.
- **Combine**—Combine welds to create a reinforced or both-sides weld.
- **Uncombine**—Convert a combine weld into independent welds.
- **Info**—Obtain information about a weld.

To Modify Welding Rod Parameters

You can modify welding rod parameters using the **WELDING RODS** dialog box.

1. Click **Welding > Rod**. The **WELDING RODS** dialog box opens.
2. Click the name of the rod to modify in the **Rod List**.
3. Modify welding rod parameters then click **Apply**.

About Lightweight Welds and Features

Welds and feature geometry are represented as quilts with a high level of complexity. You can create lightweight welds and features with fewer references and less geometry. Lightweight welds and features are especially useful for large welded assemblies. Welds and features are created without completely defining the geometry, references, and unnecessary parameters.

You can create welds and features with solid or lightweight geometry or convert them after they are

created. Lightweight welds and features are identified by a highlighted edge.

To Create Lightweight Welds and Features

Use the **WELD DEFINITION** dialog box to create lightweight welds and features.

1. Enter Assembly mode and retrieve or create an assembly.
2. Click **Applications > Welding**. The **WELDING** menu appears.
3. Define the welding environment by assigning the rod and defining welding parameters.
4. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
5. Choose the feature,weld type and combination.
6. Click the **Environment** arrow. The **Environment** dialog box opens.
7. Click the **Light** button and **OK**. The **Chain** and **Get Select** menus appear.
8. Select the chain option and the edge to create the lightweight weld or feature.
9. Click **Done**.
10. Click **Preview** to review the results or **OK** to accept.

To Convert Solid Welds and Features to Lightweight

Use the **WELD DEFINITION** dialog box to convert solid geometry to lightweight.

1. Retrieve or create an assembly with solid welds and features.
2. Click **WELDING > Create**. The **WELD DEFINITION** dialog box opens.
3. Click the **Environment** arrow. The **Environment** dialog box opens.
4. Click the **Light** button and **OK**. The **Chain** and **Get Select** menus appear.
5. Select the chain option and weld or feature to convert to lightweight geometry.
6. Click **Done** from the **Chain** menu.
7. Click **Preview** to review the results or **OK** to accept.

To Create a Welding Measurement Parameter

If measured parameters are included in the Welding Parameters table, or if you selected Measurements in the dialog box, then the system automatically invokes the Measure interface in the process of defining a weld.

1. Choose **Create** from the MEASURE PARAM menu.
2. Specify the name for the measurement parameter. If measurement parameters were specified in the Welding Parameters table, their names are listed in the CREATE MSR menu. If you want to create other parameters, choose **Enter** from the CREATE MSR menu.
3. Specify the type of measurement you want to create by selecting one of these options in the GET MEASURE menu:
   - **Edge/Crv Len**—Measure a curve or edge length.
   - **Edg/Crv Curv**—Measure a curve or edge curvature.
   - **Angle**—Measure an angle between two entities.
   - **Distance**—Measure distance between two entities.
   - **Area**—Measure an area of a surface or a quilt.
   - **Diameter**—Measure a diameter of a curved surface.
   - **Min Radius**—Measure a minimum radius of a surface.
   - **Srf Clearance**—Measure a surface clearance.
4. Select items to measure. For certain types of measurements where you need to measure between entities, specify the type of geometric reference by choosing **Point**, **Vertex**, **Plane**, **Axis**, or **Coord sys**, and then select the reference item.

5. The system informs you that the measurement parameter has been created; the current value of this parameter is given in parentheses.

Other Welding Measurement Operations

Commands for working with measurements are listed in the MEASURE PARAM menu:

- **Create**—Create a new measurement.
- **Delete**—Delete an existing measurement.
- **Redo**—Redo a measurement. Select the name of the measurement to be redone and proceed to create a new measurement.
- **Info**—Obtain information about measurements that exist in the weld.
- **Show**—Show references for a selected measurement. After you select a measurement name from the namelist menu, the referenced entities are highlighted.

Tip: Entering Information in the Welding Parameters Table

To set up a relation between a desired welding parameter and measurements of some geometric elements, you enter the following information in the Welding Parameters table.

1. Enter an equation as a value for the desired parameter.
2. For the measurement parameter used in the relation, enter the following line:

   parameter_name `measure`

   where:

   parameter_name — is the name of the parameter.

For example, the following lines can be added to the Welding Parameters table:

```
X_SECTION_AREA a*b/2*1.2
a measure
b measure
```

Note: The length of a relation is limited to a single line of text; conditional statements are disallowed.

To Control the Cross-Section of the Weld Using Relations

If you want the X_SECTION_AREA parameter to update whenever a certain geometry changes, you can specify a relation that causes this parameter to be recalculated automatically.

Two types of parameters can be used in a relation that you specify for the X_SECTION_AREA parameter:

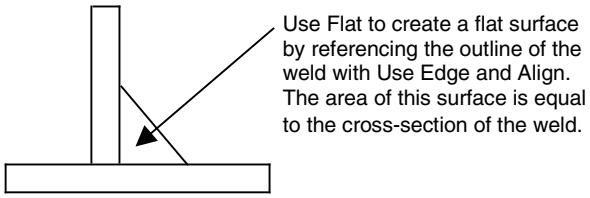
- Model dimensions or any dimensions of an existing weld (such as, leg size) in their symbolic form (such as, d32).
- Measurement parameters.

When you access the Welding Parameters Info window, the relation that you entered for the X_SECTION_AREA parameter is listed in the table. Values of the measurement parameters can be viewed in the Model Info window.

Note: Alternatively, you can set up a relation to automatically update the X_SECTION_AREA parameter whenever the area of the cross-section of the weld changes.

1. Create a weld and specify the initial value for the X_SECTION_AREA parameter.
2. Create an assembly surface feature using the **Flat** option with the profile of the surface corresponding to the cross-section of the weld.

Creating a Surface Based on the Weld's Cross-Section



3. Create an assembly Evaluate feature, measuring the area of this cross-sectional quilt. Specify the name of the Evaluate feature (such as, X_SECTION_AREA).
4. Create an assembly relation which relates the measurement of the surface feature and the X_SECTION_AREA parameter of the weld. (Obtain the internal ID of the weld using **Info/Feat Info**.) Enter the following relation:
   `X_SECTION_AREA:fid_weld_id = X_SECTION_AREA:fid_evalfeat_id`
   where:
   weld_id - is the internal feature ID of the weld.
   *evalfeat*_id - is the internal ID or the name of the Evaluate feature.
5. Regenerate the model. The system updates the X_SECTION_AREA parameter of the weld; the volume of the weld updates accordingly.
6. If desired, you can put the flat surface on a layer and blank this layer.

To Create a Compound Weld

1. Choose **Combine** from the WELD UTILS menu.
2. Specify how you want to combine welds by checking options in the COMBINE OPTS menu, followed by **Done**.
3. Select the welds to be combined. You can pick the welds with the **Pick** option, or select their names from the namelist menu with the **Sel By Menu** option.

With Pro/WELDING, you can automatically show welding symbols for certain types of compound welds. The following is a list of compound welds for which symbols are supported.

Reinforced welds (with Fillet always being a reinforced weld):

- Square Groove
- Bevel Groove
- Flared Bevel Groove
- J Groove

Both sides welds:

- Fillet
- Square Groove
- V Groove
- Bevel Groove
- U Groove
- J Groove
- Flared V Groove
- Flared Bevel Groove

About Compound Welds

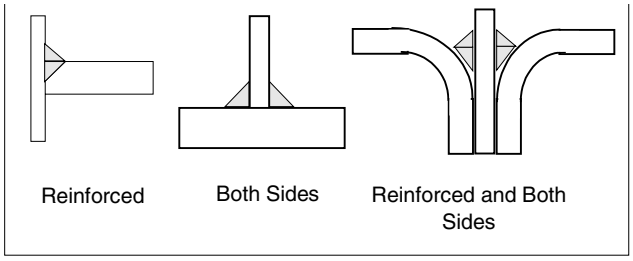
There are two types of compound welds listed in the Combine Opts menu (see the following figure). These

are checkmark options, and they can be selected alternatively or in combination.

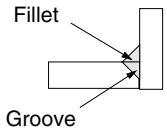
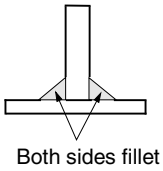
- **Reinforced**—Combine two welds to create a reinforced weld.
- **Both Sides**—Combine two welds on both sides of the joint into one weld.

You can combine up to four individual welds into a compound weld.

Possible Combinations of Compound Welds



Note: All reinforced welds can also be both-sided, but not all both sides are reinforced.

Type of Compound Weld	Example	Description
Reinforced		A V groove and fillet are combined in a single welding sequence.
Both sides		Two fillets are combined in a single welding sequence.

To Uncombine a Compound Weld

The welds included in a compound weld can be uncombined with the **Uncombine** command in the Weld Utils menu. This causes all associated welds in the combined weld to become independent, and each weld restores its own sequence ID.

1. Choose **Uncombine** from the WELD UTILS menu.
2. Using the **Sel By Menu** option, choose the name of the compound weld from the namelist menu. The combined weld is now converted into individual welds.

To Blank or Unblank Welds

1. Choose **Set Display** from the WELDING menu.
2. To blank welds, choose **Blank** from the WELD DISP menu and specify welds to be blanked. You can indicate welds using options in the SPECIFY BY menu.

To unblank welds, choose **Unblank** from the WELD DISP menu and specify welds to be unblanked.

To Suppress Selected Welds

- **Suppress**—Suppress selected welds. Select a weld to be suppressed and choose **Done** from the SELECT FEAT menu.

To Resume Selected Suppressed Welds

- **Resume**—Resume selected suppressed welds. Choose the names of the welds to be resumed by selecting from the namelist menu.

To Reorder the Sequence of Welds

- **Reorder**—Re-arrange the sequence of welds. This is done in the same manner as with other Pro/ENGINEER features.

To Pattern a Spot Weld

A spot weld can be patterned as other Pro/ENGINEER features using the dimension pattern functionality.

1. Choose **Pattern** from the WELD UTILS menu.
2. Select a datum point referenced by the spot weld.
3. The PAT DIM INCR menu appears. Create a pattern of datum points (these points are used to create a pattern of welds).
4. Once the pattern of datum points is created, choose **Pattern** again, and this time pick the weld. A pattern of spot welds are created.

To Delete a Pattern of Welds

You can delete a pattern of welds using the **Del Pattern** command. This deletes all patterned welds with the exception of the original.

1. Choose **Del Pattern** from the WELD UTILS menu.
2. If you want to delete all patterned welds, including related datum points, pick a datum point from the pattern.
   ... or...
   If you want to delete a pattern of welds without deleting datum points, pick the weld feature from the pattern.

To delete all welds in the pattern, including the original weld, use the **Delete** command. If you pick on a datum point from the pattern, the entire pattern of welds with datum points are deleted. If you pick on a weld in the pattern, only the welds are deleted.

To Modify the Number of Welds in a Pattern

You can modify the number of welds in the pattern by modifying the pattern parameter with the Mod Dim command in the Weld Utils menu.

1. Choose **Mod Dim** from the WELD UTILS menu.
2. Pick one of the datum points referenced by the pattern of welds. This displays the pattern parameter (such as *4 datum points*).
3. Pick the pattern parameter and enter its new value.
4. When you exit the WELD UTILS menu, the system updates the pattern.

To Delete Welds

1. Chose **Delete** from the WELD UTILS menu.

2. Select welds to be deleted.
3. Choose **Done** from the SELECT FEAT menu.

To Redefine a Weld

You can redefine a weld with the **Redefine** command in the Weld Utils menu. Redefining a weld involves changing any of the elements that you defined at weld creation.

1. Choose **Redefine** and select a weld to be redefined.
2. A dialog box appears, listing elements of the weld feature.
3. Select an element that you want to redefine and choose the Define button in the dialog box. Redefine the element using the same menu options and techniques as when you defined it initially.
4. Continue the process by selecting another feature element and redefining it.
5. Once you have made all required changes, choose **OK** from the dialog box.

To Modify the Dimensions of a Weld

- **Mod Dim**—Modify weld dimensions.

To Assign a Different Welding Rod to a Weld

1. Choose **Modify** from the WELDING menu.
2. Choose **Change Rod** from the MOD WELD menu.
3. Select a weld to have its rod re-assigned.
4. From the ROD SEL menu, select a rod that you want to assign to the selected weld.

To Modify the Parameters of a Weld

1. Choose **Modify** from the WELDING menu.
2. Choose **Mod Params** from the MOD WELD menu.
3. Select a weld whose parameters you want to modify.
4. The WELD PARAMS menu appears.

To Regenerate a Weld

After you have modified dimensions of the weld using **Mod Dim** in the MOD WELD menu or dimensions of the assembly, regenerate the model with the **Regenerate** command in the WELDING menu.

The Regenerate command works as with other assemblies.

About Weld Symbols in Drawings

The standard Pro/ENGINEER welding symbol libraries are designed to provide general ANSI and ISO symbols. The system uses the symbols in the system weld-symbol libraries when it is automatically placing weld symbols. However, since many organizations have their own preferences for defining symbols, the symbols can now be easily customized.

> **Note:** When Pro/ENGINEER is automatically placing welding symbols on a drawing, it recognizes *only* those welding symbol names that are included in the standard Pro/ENGINEER welding-symbol libraries. Therefore, if you create a new welding symbol "from scratch," you must use it to replace an *existing* symbol in the welding-symbol library.

As a general rule, it is safer to redefine an existing weld symbol. The procedures are described in the following section.

To Redefine a Weld Symbol

1. Choose DETAIL > **Create** > **Symbol** >**Definition** >**Symbol** > **Retrieve**. The OPEN dialog box appears.

2. Choose a symbol from the System Syms (system symbols) library.
3. You can do any or all of the following:
   - Add as many copies of variable texts as you want.
   - Change the default values of variable texts.
   - Add and delete as many notes and entities as you want and place new ones in any group (or in no group at all).
   - Redefine the cosmetics of existing notes and entities.
   - Move the origin position for Left Leader and Right Leader. or add other leader types.
   - Add parameters to the symbol definition.

When you are redefining weld symbols, the following restrictions apply:

- All of the groups that existed in the original definition *must* remain in the new definition. You cannot add new groups or change the names of existing ones.
- If you add new variable text or change the name of an existing piece of variable text, the new name *must* be the same as that of an existing variable text in the original.
- The height type of the symbol instance *must* be the same in the new symbol as it was in the original.
- The **Left Leader** and **Right Leader** placement types must *both* exist in the new weld symbol.

Tip: Redefining Weld Symbols in Drawings

Before you start, you should specify the user's symbol root directory, because Pro/ENGINEER will only let you save the redefined weld symbols in or below it. Later, you can copy the redefined symbols over the old ones in the system symbol libraries.

To Store a Redefined Weld Symbol

1. Choose DWG SYMBOL> **Write**.
2. Specify the symbol by choosing **Name** or **Pick Inst** from the GET SYMBOL menu.
3. Enter the name of the destination directory (offset from the user's symbol root directory).
   Note: You do not have to store a symbol on disk in order to continue using it in a drawing. However, if you don't, the system only stores the symbol locally in the drawing currently in session and does not make it available for use in other drawings or by other users.

To Replace a Symbol in a System Weld Symbol Library

The system weld-symbol libraries are located in the installation directory path <install_dir>/symbols/library_syms/weldsymlib. To replace a standard symbol with one that you have redefined, ask your system administrator to do the following:

1. Move the original (system-supplied) symbol from the system weld library to another directory or rename it.
2. Copy the new user-redefined symbol into the system weld library.

To Display Welding Symbols for Existing Welds in a Drawing

1. Choose **Detail** from the DRAWING menu.
2. Choose **Show/Erase** from the DETAIL menu. The Show/Erase dialog box appears.
3. Click the **Symbol** in the TYPE box.
4. Select an option from the SHOW BY menu:
   - **Feature**—Show a symbol for a selected weld feature.
   - **Feature & View**—Show a symbol for a selected weld in a selected view.

- **Part**—Show all symbols in a selected component.
- **Part & View**—Show all symbols in a selected view of a selected component.
- **View**—Show all symbols in a selected view.
- **Show All**—Show all the symbols in the drawing. Each welding symbol is displayed only once.

Welds Supported in ISO

When welding symbols are shown in drawings, you can choose between the ANSI and ISO standards with the detail setup option, weld_symbol_standard. This option has two values, STD_ANSI (the default) and STD_ISO.

The welds supported in ISO are:

- Non-grooved
  - Fillet
  - Plug
  - Slot
  - Spot
- Grooved
  - Square
  - Bevel
  - V
  - U
  - J

  Note: For bevel and V symbols: if root_open is more than zero, the "steep" version of the symbol is used. If prep_depth is less than material thickness, the broad version is used.

To Obtain Information on Welds

1. Choose **Weld** in the WELD INFO menu.
2. Select welds for which you want to obtain information.
3. The system displays an Information Window. and creates a file, weldinfo.dat, in the working directory.

Tip: Another Way To Obtain Information on Welds

You can use the **Info** option in the Weld Utils menu to open an Information Window, listing information about a selected weld.

To Obtain Information on Weld Parameters

1. Choose **Parameters** in the WELD INFO menu.
2. Select a weld for which you want to obtain parameter information.
3. The system displays an Information Window with the table of weld parameters.

About Welding Parameters

To access welding report parameters, choose **weldasm** from the Report Sym menu.

Welding Pro/REPORT Parameters

Parameter Name	Definition
&weldasm.weld.seq_id	Lists the ID of a welding sequence.

&weldasm.weld.type	Lists the type of welding sequence (such as, fillet, groove, and so on).
&weldasm.weld.len	Lists the length of a weld (in assembly units).
&weldasm.weld.size	Lists the size of a weld in assembly units according to its type: Fillet: L or L1xL2 Groove: Penetration + Root Penetration Plug/slot: Depth + Root Penetration Spot: Diameter
&weldasm.weld.volume	Lists the volume of a weld (in assembly units).
&weldasm.weld.rodlength	Lists the length of the rod used by a weld (in rod units).
&weldasm.weld.timeused	Lists time used to finish routing a weld (in hours).
&weldasm.weld.*User-defined*	User-defined weld parameters. *User-defined* - is the parameter name.
&weldasm.weld.rod.name	Lists the name of the rod used by a weld.
&weldasm.rod.name	Lists the rod name.
&weldasm.rod.totallength	Lists the total length of a rod in an assembly.
&weldasm.rod.totalmass	Lists the total mass of a rod in an assembly.
&weldasm.rod.*User-defined*	User-defined rod parameters. *User-defined* - is the parameter name.
&weldasm.totallength	Lists the total length of all welds in the assembly (in assembly units).
&weldasm.totalmass	Lists the total amount of rod mass used in the assembly (in rod units).
&weldasm.totaltime	Lists total welding time in the assembly (in hours).

A Sample Pro/REPORT Table

Welding Sequence	Weld Type	Rod Name	Weld Length
weldasm.weld.seq_id	weldasm.weld.type	weldasm.weld. rod.name	weldasm.weld.len

Pro/REPORT Information Table

Welding Sequence	Weld Type	Rod Name	Weld Length
1	Groove	Steel_Rod1	.4
2	Fillet	Steel_Rod1	.3
3	Fillet	Steel_Rod1	.4
4	Groove	Steel_Rod2	.2

To Obtain Information on Weld/Rod Length

1. Choose **Length** in the WELD INFO menu.
2. From the WLD INF TYP menu, select one of the options:
   - **Sel Weld** — Obtain length information for selected welds. An Information window appears with the Length Information table. In addition, the contents of the table are stored in a file called *weldlengthinfo.dat.#* in the current working directory.
   - **Sel Rod**—Obtain length information for specified rods used by different welds in the assembly.
3. Select **Rod Names** from the ROD NAMES menu. An Information window appears with the Length Information table. In addition, the contents of the table are stored in a file "rodlengthinfo.dat.#" in the current working directory.

   Note: *Total Length* is the sum of the lengths of the specified rods. *Length* is computed by summing the lengths of the particular rod used by all the welds in the assembly.

To Obtain Information on Weld/Rod Mass

1. Choose **Mass** in the WELD INFO menu.
2. From the WLD INF TYP menu, select one of the options:

- **Sel Weld**—Obtain mass information for selected welds. An Information window appears with the Mass Information table. In addition, the contents of the table are stored in a file "weldmassinfo.dat.#" in the current working directory.
- **Sel Rod**—Obtain mass information for specified rods used by different welds in the assembly.

  Select rod names from the ROD NAMES menu. An Information window appears with the Mass Information table (see the following figure). In addition, the contents of the table is stored in a file "rodmassinfo.dat.#" in the current working directory.

Note: *Total Mass* is the sum of the mass of the specified rods. *Mass* is computed by summing the mass of the particular rod used by all the welds in the assembly.

To Obtain a Bill of Materials

If you select BOM from the Weld Info menu, a BOM table generates.

A Sample BOM Table

```
                BOM  Information

Total Length (assembly units) 1696.1000
Total Mass (rod units)        4.6576
Total Welding Time(Hours)     169.6100


Seq ID  Type     Rod     Length    Mass   Time
-------------------------------------------------
 1    Groove    STEEL_ROD 239.22    0.66   23.92
   REINFORCED BY
 *    Fillet    STEEL_ROD 239.22    0.66   23.92

 2    Fillet    STEEL_ROD 239.22    0.66   23.92
 3    Fillet    STEEL_ROD 500.00    1.37   50.00
 4    Groove    STEEL_ROD 239.22    0.66   23.92
   BOTH SIDES WELD
 *    Groove    STEEL_ROD 239.22    0.66   23.92



Rod     Length    Mass
-----------------------------------------------
STEEL_ROD   0.03    4.66
```

About Model Tree

You can display a graphical hierarchy of a manufacturing model in the form of a Model Tree window. When you create or retrieve a manufacturing model, the system displays the Model Tree window.

Using the right mouse button, click to view the following Model Tree options:

- **Open**—Open the feature in another display screen.
- **Delete**—Delete the feature.
- **Suppress**—Suppress the feature.
- **Modify**—Modify the feature.
- **Redefine**—Open the **Component Placement** dialog box to redefine the feature.
- **Reroute**—Open the **Reroute Refs** menu to reroute the feature.
- **Replace**—Open the **Replace Component** dialog box to replace the feature manually or by layout.
- **Ref Control**—Determine scope of components to be referenced.
- **Fix Location**—Constrain location of the components.
- **Feature Create**—Create a feature.
- **Component**
  - Create—Open the **Component Create** dialog box to create a component.
  - Assemble—Assemble components.
  - Include—Include components.
- **Note Create**
  - Assembly—Create a note for the assembly.
  - Component—Create a note for the component.
- **Info**
  - Parent Child—Display the Parent/Child relationship.
  - Feat Info—Display feature information.
  - Model Info—Display model information.

About Welding Process Parameters

You can create and modify welding rod and process parameters by using the **Welding Processes** dialog box. In addition, optional and user defined parameters can be created or modified. A welding rod provides the welding material necessary to create a weld bead. Weld process information includes parameters and weld finishing information.

Within your welding assembly, each rod is defined by its name and parameters associated with this rod. Rod parameters are stored with the model. If you want parameters of a particular rod to be used for defining welds in other assemblies, you can store the rod parameters to a file on disk.

The Welding Process dialog box consists of the following options:

- New – Create a new welding process.
- Open – Read a new welding process from the disk.
- Save – Save a welding process to a disk.
- Save As – Save a welding process to a disk with a different name.
- Delete – Delete selected weld process.
- **Utilities**
  - Set as Default – Set selected process as a default process.
  - Assign – Assign selected welding processes to welding features.
- **Process Parameters**
  - Process Name – Name of weld process.

- Machine Type – Select manual or robotic type of machine.
- Treatment – Select Low Hydroge, Pre-Heating, Post-Heating, or no treatment.
- Feedrate – Specify welding rod feed rate.
- Specification – Type specification number.
- Max Allowed Length – Maximum allowed welding rod length.
- Min Allowed Length – Minimum allowed welding rod length.
- Max Root Opening – Maximum root opening.
- Min Root Opening – Minimum root opening.

- **Field Weld** – Indicate that a weld is made at a location other than during initial construction of the assembly. The field weld symbol is denoted by a filled black circle at the junction between the arrow and the reference line.
- **Finishing** – Choose the shape or contour of the weld surface.
- **Optional and User Defined Parameters** – Define name and value of user defined parameters.

To Create Welding Process Parameters

You can create or modify welding process parameters using the **WELDING PROCESSES** dialog box.

1. Click **Welding > Process**. The **Welding Processes** dialog box opens.
2. Click **File > New**. Enter the name of the welding rod then click  . The name of the welding rod appears in the process list.
3. Set the welding process parameters.
4. Choose the shape or contour of the weld surface.
5. Determine optional and user defined parameters.
6. Click **Apply.** Pro/Welding issues a message indicating the welding rod with the specified name has been successfully created.

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